



A Verification Procedure for MSC/NASTRAN Finite Element Models

*Alan E. Stockwell
Lockheed Engineering & Sciences Company • Hampton, Virginia*

National Aeronautics and Space Administration
Langley Research Center • Hampton, Virginia 23681-0001

Prepared for Langley Research Center
under Contract NAS1-19000

June 1995

Printed copies available from the following:

NASA Center for AeroSpace Information
800 Elkridge Landing Road
Linthicum Heights, MD 21090-2934
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 487-4650

Acknowledgments

The author wishes to acknowledge the contributions of Mercedes Reaves and Raymond Kvaternik of NASA Langley Research Center. Both Ray and Mercedes offered several helpful suggestions, and Ray supplied a stack of related publications from his personal files.

Table of Contents

Acknowledgements	iii
1.0 Introduction	1
2.0 Preprocessor Checks	1
3.0 Analytical Checks	2
4.0 Model Verification Procedure	9
5.0 Summary	17
6.0 References	18

PRECEDING PAGE BLANK NOT FILMED

1.0 Introduction

Finite element models (FEMs) are used in the design and analysis of aircraft to mathematically describe the airframe structure for such diverse tasks as flutter analysis and actively controlled landing gear design. FEMs are used to model the entire airplane as well as airframe components. Model verification procedures are especially important for large-scale FEMs of an entire airplane which are developed by an outside contractor or agency and are used for aeroelastic and dynamic analyses. Since there is no test data to validate the FEMs during the preliminary design stage, it is especially important for both model developers and users to understand the limitations of the models and to ensure that they are used correctly. The purpose of this document is to describe recommended methods for verifying the quality of the FEMs and to specify a step-by-step procedure for implementing the methods. The procedure has been successfully applied to large-scale FEMs of preliminary design concepts for the NASA High Speed Civil Transport (HSCT) aircraft.

The document is divided into four sections. Section 1 is the Introduction. Section 2, Preprocessor Checks, briefly describes suggested procedures for checking a model using a graphical preprocessor. Section 3, Analytical Checks, describes methods of verifying the mathematical correctness of the model using the MSC/NASTRAN finite element code. Section 4, Model Verification Procedure, presents a step-by-step procedure for implementing the analytical checks described in Section 3. Section 4 is intended to be a working document containing a "cookbook" procedure to facilitate the model checking process and help ensure a consistent level of quality. Although this procedure may uncover modeling errors, it is not an exhaustive investigation of modeling details, nor does it address the issue of whether the structure is modeled appropriately. The assumption is made that the structure was carefully modeled by the developer. The FEM user's task is to ensure only that the model makes mathematical sense and contains no obvious errors or omissions.

Several methods, such as kinetic energy and effective mass, can be used to evaluate the dynamic properties of FEMs. These methods can also identify weaknesses or modeling errors. However, their primary purposes are to characterize the dynamics of the structure and to guide a dynamicist in selecting a valid set of structural vibration modes for a particular analysis. A discussion of these methods is beyond the scope of this document.

2.0 Preprocessor Checks

2.1 Introduction

Preprocessor model checks are important in developing a FEM. However, some of the standard checks employed in the model development process (e.g., element aspect ratio or taper) may not be necessary or appropriate when a model created by another organization is being verified. The discussion that follows focuses on those particular preprocessor checks which are valuable tools in verifying a model developed outside the user's organization.

2.2 Visual Checks

An analyst using a FEM developed by an outside source can employ preprocessor visual checks to become familiar with the model and to ensure that it looks reasonable. Preprocessor plots of the model allow the analyst to verify the overall shape of the model, as well as key dimensions. If more than one coordinate system has been used to define model geometry, the plots are an excellent method of determining whether key structural details are oriented correctly. Most preprocessors allow the user to group grid points and elements according to criteria such as physical or material property number. Although it may not be feasible to check every property in a model, the plots offer a quick method of checking selected data. Plots of loads and boundary conditions can also be used to quickly check that they are applied correctly.

It should be recognized that every time a translation is made from one analysis or preprocessor code to another (e.g., PATRAN [1] to NASTRAN [2], NASTRAN to I-DEAS [3], etc.), there is a potential for introducing errors. The analytical checks described in Section 3 provide a good basis for ensuring that the results of some of the preprocessor checks are still valid after the model has been translated into an MSC/NASTRAN input file.

2.3 Element Checks

Most preprocessors will perform element distortion checks that measure quantities such as taper, skew angle and aspect ratios. Modeling details that violate generally accepted guidelines may not necessarily be incorrect. However, it is useful for an analyst to be aware of the expected quality of results obtained from various parts of the model.

Weight property checks may also be useful. Differences in the results between the analysis program and the preprocessor may indicate translation problems.

2.4 Summary

This section has presented some brief guidelines for using a graphical preprocessor to verify a model. A detailed discussion of commercial FEM preprocessors is beyond the scope of this document, and it is left to the individual analyst to correctly use the features of a selected preprocessor in an effective manner.

3.0 Analytical Checks

3.1 Introduction

The purpose of the analytical checks described in this section is to ensure the mathematical soundness of the model and to uncover any gross modeling errors, such as missing elements or incorrectly applied boundary conditions. The material presented herein consists of generally accepted practices and procedures. Most of the methods are described in references 4, 5 and 6.

3.2 Pre-analysis Mass, Stiffness and Matrix Reduction Checks

Several analytical model checks can be performed prior to any static or dynamic analysis. These checks are referred to as "pre-analysis" checks, because they are computations which are performed on the mass and/or stiffness matrices and are independent of specific boundary conditions or applied loads.

3.2.1 Constraint Checking

MSC/NASTRAN [2] provides the option in any of the Structured Solution sequences (SOL 101-200) of requesting a "superelement checkout" run by including "PARAM,CHECKOUT,YES" in either the Bulk Data Deck or Case Control Deck. The "checkout" option triggers a series of checks in the "Bookkeeping and Control" (Phase 0) subDMAP. The run is automatically terminated before the matrix assembly, generation and reduction operations begin in the Phase I subDMAP. While this option is primarily intended for checking superelement models, it includes a sequence of multipoint constraint checks which are useful for any model. These checks detect the presence of internal constraints (grounding) and ill-conditioning. The checks operate on the multipoint constraint equation matrix, R_{mg} , formed in module GP4 from the MPC and rigid body Bulk Data entries. In order to perform the checks, NASTRAN partitions R_{mg} into dependent (m-set) and independent (n-set) sub-matrices, i.e., $R_{mg} = [R_{mm} \ R_{mn}]$. As described in Section 9.4.1 of reference 2, three tests are performed:

1. A matrix of rigid-body vectors, U_{gh}^0 , is assembled using the VECPLOT module, and the product

$$E_{mh} = R_{mg} U_{gh}^0$$

is calculated. The terms of E_{mh} larger than PARAM,TINY are printed. These terms usually indicate internal constraints, although exceptions to this may occur if there are MPC equations involving scalar points. A simple example of an internal constraint is an MPC equation involving two degrees of freedom, in which the coefficient for the independent degree of freedom is inadvertently left blank. NASTRAN will assume that the coefficient is zero, thereby grounding the dependent degree of freedom.

2. The product $R_{mm}^g = R_{mg} R_{mg}^T$ is calculated and decomposed by the DCMP module. During the solution process, NASTRAN decomposes symmetric structural matrices into upper and lower triangular factors and a diagonal matrix, e.g.,

$$K = L D L^T$$

where L is the lower triangular factor and D is called the "factor diagonal matrix." Note that the upper triangular factor for a symmetric matrix is equal to LT. Symmetric decomposition, followed by forward/backward substitution, is a computationally efficient alternative to matrix inversion. An additional benefit of decomposition in MSC/NASTRAN is the diagnostic messages that alert the user to problems in the matrices. Each diagonal term of K is divided by the corresponding term of the factor diagonal matrix, D, and ratios larger than PARAM,MAXRATIO are printed. The number and location of any negative terms in

the factor diagonal matrix are also printed. In the case of the constraint matrix, R_{mg} , the terms flagged by the decomposition of R_{mm}^T indicate the presence of linearly dependent rows in R_{mg} , i.e., redundant constraints. This condition will probably cause singularities or poorly conditioned constraints if the problem is not corrected.

3. The product $R_{mm}^T = R_{mm}R_{mm}^T$ is calculated and decomposed, and factor diagonal terms larger than MAXRATIO are printed. The results of this check may be compared to the results of step 2. A degree of freedom flagged here that was not flagged in step 2 indicates that a problem exists in the dependent partition, R_{mm} , but not in the matrix containing all DOF (R_{mg}). Therefore, an error was made in specifying the dependent degrees of freedom.

The "checkout" option automatically stops the solution process after the constraint checks. Additional model verification may be accomplished by using either specially-developed Direct Matrix Abstraction Programs (DMAPs) or static analyses.

3.2.2 Grid Point Weight Generator (GPWG)

After assembling the mass and stiffness matrices, NASTRAN can print out a summary of the structure's weight properties, including center of gravity, total weight, and inertia matrix. These checks are performed before the mass matrix is converted from weight units to mass units. The user should check the location of the center of gravity, the total weight in each direction, the principal mass axis directions and the inertia matrix. The weight should be the same in all three directions unless scalar masses are used. Note that the inertia matrix, $I(s)$, is not in tensor form. The off-diagonal terms of $I(s)$ must be multiplied by -1.0 to convert the matrix to tensor form. The inertia tensor is referred to in the MSC/NASTRAN documentation [7] as the "intermediate inertia matrix, \tilde{I} ".

In the sample GPWG output shown below, the reference point is taken as grid point 150002 in the model. Therefore the rigid-body mass matrix, MO, represents the mass properties of the structure with respect to grid 150002, not the origin of the basic coordinate system, which is the default. The center of gravity is also calculated with respect to the reference point.

```

O U T P U T   F R O M   G R I D   P O I N T   W E I G H T   G E N E R A T O R
REFERENCE POINT = 150002
M O
* 3.719099E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 *
* 0.000000E+00  3.719099E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.179047E+03 *
* 0.000000E+00  0.000000E+00  3.719099E+00  0.000000E+00  -2.179047E+03  0.000000E+00 *
* 0.000000E+00  0.000000E+00  0.000000E+00  4.126100E+04  0.000000E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00  -2.179047E+03  0.000000E+00  2.125744E+06  0.000000E+00 *
* 0.000000E+00  2.179047E+03  0.000000E+00  0.000000E+00  0.000000E+00  2.166884E+06 *
S
* 1.000000E+00  0.000000E+00  0.000000E+00 *
* 0.000000E+00  1.000000E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00  1.000000E+00 *

DIRECTION
MASS AXIS SYSTEM (S)      MASS          X-C.G.        Y-C.G.        Z-C.G.
X                      3.719099E+00  0.000000E+00  0.000000E+00  0.000000E+00
Y                      3.719099E+00  5.859073E+02  0.000000E+00  0.000000E+00
Z                      3.719099E+00  5.859073E+02  0.000000E+00  0.000000E+00

```

```

          I (S)
* 4.126100E+04 0.000000E+00 0.000000E+00 *
* 0.000000E+00 8.490239E+05 0.000000E+00 *
* 0.000000E+00 0.000000E+00 8.901639E+05 *

          I (Q)
* 4.126100E+04 *
* 8.490239E+05 *
* 8.901639E+05 *

          Q
* 1.000000E+00 0.000000E+00 0.000000E+00 *
* 0.000000E+00 1.000000E+00 0.000000E+00 *
* 0.000000E+00 0.000000E+00 1.000000E+00 *

```

The Grid Point Weight Generator is described in detail in reference 6.

3.2.3 Grid Point Singularities

MSC/NASTRAN's Grid Point Singularity Processor (GPSP) is automatically included in all solution sequences. Grid point singularities are defined as zero or near zero terms in the stiffness matrix. They are either the result of modeling errors such as missing elements, or are caused by undefined DOF, such as in-plane ("drilling") rotations of plate elements. The GPSP module inspects each DOF in the N-set (independent degrees of freedom left after multipoint constraint elimination). Principal stiffnesses are calculated for the three translational and three rotational DOF at each grid point, and each grid point stiffness term is divided by the corresponding principal stiffness. The resulting ratio, ϵ , is compared to the minimum allowable value set by the user with PARAM,EPZERO (default = 10^{-8}). Singular degrees of freedom, i.e., DOF whose ϵ is less than EPZERO, are listed along with their corresponding "stiffness ratio". If the parameter AUTOSPC is set to YES (default), the DOF are automatically constrained and the original set membership and new set membership (after the DOF are constrained) are listed. The GPSP output should be carefully inspected. Reference 6 contains a thorough discussion of the procedure.

3.2.4 Mechanisms

Mechanisms lead to stiffness matrix singularities involving two or more grid points. An example of a mechanism is a section of a structure that is capable of rigid-body motion in one or more directions. MSC/NASTRAN automatically checks for mechanisms every time it performs a decomposition using the DCMP module. Diagnostics are printed if any mechanisms are detected. Unlike the one-time grid point singularity check, decompositions of various matrices are performed in several places in a typical solution. For example, if the user requests static condensation, the static transformation matrix, GOAT, which relates the omitted degrees of freedom to the analysis degrees of freedom, is obtained by a decomposition of the stiffness matrix, KOO. The DCMP module is used again in a typical static analysis in which the solution is obtained by a decomposition of the "leftover" stiffness matrix, KLL, followed by a forward-backward substitution. During decomposition each diagonal term in the stiffness matrix is compared to the corresponding term of the factor diagonal matrix, D. Ratios larger than the value set by the user parameter MAXRATIO are printed, with the corresponding grid point and DOF identified. The user should carefully inspect the output file. If there are decomposition diagnostics, determine the set level (A-set, L-set, etc.) at which the problems occurred. A more detailed discussion of mechanisms is contained in references 6, 8 and 9.

3.2.5 Multi-Level Strain Energy Checks

Multi-level strain energy checks are another means of detecting modeling errors. The checks are not built into the structured solution sequences, however a DMAP alter is available. The alter, "checka.v68", is located in the "misc" subdirectory on the MSC/NASTRAN delivery tape. The checks are referred to as "multi-level" because the computations are performed at various set levels such as the G-set, N-set, and A-set level. Since each level represents a significant step in the reduction process leading to the final equations, the checks can be a useful means of determining both the location and the cause of errors. For example, an error that occurs at the A-set level, but does not occur at the N-set level, would be caused by a static or dynamic reduction error rather than a rigid-body constraint problem.

An important issue to consider when using these checks is the application of specific boundary conditions. If all single point constraints except those that involve zero-stiffness DOF (e.g., drilling rotations in QUAD4 plate elements) are removed, then the checks will be independent of specific boundary conditions. This may be useful in identifying hidden problems, such as grounding. However, the analyst may also want to check the effects of the single point constraint (SPC) elimination (See the discussion in section 3.2).

The multi-level strain energy check procedure consists of computing a set of six rigid-body displacement vectors, then using them to compute forces, rigid-body strain energy, and rigid-body mass matrices. The checks proceed as follows:

Stiffness Checks:

1. At the G-set level, compute a set of rigid-body vectors, RBG, using the VECPLOT module.
2. Compute the reaction forces resulting from the rigid-body motion, and print the normalized non-zero forces:

$$REACG = KGG * RBG$$

3. Compute and print the strain energy:

$$CHKKGG = RBG^T * REACG$$

4. Repeat steps 1 through 3 at the N-set level (the N set contains all independent DOF not eliminated by multipoint constraints).
5. Repeat steps 1 through 3 at the A-set level. To obtain the A-set stiffness matrix, NASTRAN first partitions the N set into the F set (free DOF) and the S set (DOF eliminated by single point constraints). The F set is then partitioned into A set (analysis set) and O set (omitted) DOF, and a reduced stiffness matrix, KAA, is typically computed by the process of Guyan (static) reduction, generalized dynamic reduction, or component mode synthesis. If no reduction is requested by the user, the F set and A set will be equivalent.

The diagonal terms of the strain energy matrix should all be nearly zero if there are no errors (such as grounding problems) in the stiffness matrix. The reaction forces are normalized by dividing each term by the largest term in the vector. If there are non-zero terms, the elements of the normalized reaction force matrix (REACGNRM, REACNNRM, or REACANRM) can be surveyed to find the largest forces.

Mass Checks:

Similar calculations are made by pre- and post-multiplying the mass matrices by the rigid-body vectors. This process results in a 6x6 matrix that can be compared to the rigid-body mass matrix, MO, calculated by the Grid Point Weight Generator. For example, at the G-set level,

$$WGHT = RBG^T * MGG * RBG$$

The matrix WGHT should be equal to MO. The matrix WGHTN (calculated at the N set level) should also be equal to MO. However, when the same calculations are performed at the A-set level, the matrix WGHTA will not necessarily be the same if the structure has any single point constraints applied. Any mass associated with the constrained DOF will not be accounted for in WGHTA. This problem can be avoided by temporarily removing the SPC card in the Case Control Deck. The mass checks are a useful means of verifying that the proper mass of the structure was retained throughout the reduction process.

3.3 Static Analysis Checks

Simple static analyses can be performed to check a model. If pre-analysis checks have been made, some of the static analyses may be somewhat redundant. However, a static analysis is generally a relatively inexpensive means of checking the soundness of a finite element model.

3.3.1 1-g Check

Using an appropriate set of boundary conditions, apply a unit gravity load to the entire structure. The resulting displaced shape can be inspected for "reasonableness." For example, are there any parts of the structure that show suspiciously large displacements? Does the overall deformed shape look reasonable?

3.3.2 Enforced Displacement Check

Constrain a single grid point in all six DOF, release all other single-point constraints which represent a physical connection to ground, and apply an enforced displacement at each restrained DOF, one at a time. For each of the six resulting analyses, check the deformed shape both visually, and numerically. For example, if a unit displacement is applied in the x direction, then all x displacements should be equal to 1. For large structures this check is easier to assess by using a post-processor to display the deformed shape.

The issue of boundary conditions is again an important consideration in this process. The procedure calls for checking the structure in the free-free (completely unrestrained) condition. However, for some structures (e.g., symmetric structures) the analyst may also

want to check the structure with SPCs applied to insure that strain-free motion is still possible in the DOF that are not affected by the boundary restraints.

3.3.3 Checks Against Reference Data

Frequently there exists analytical or experimental data which can be used to validate the model. For example, if a model is delivered from one contractor to another, then translated from one analysis program to another, the results of the translation can be checked if a set of reference data is available. This might take the form of a set of displacements or element forces caused by a given loading. It is a relatively simple task to make such a comparison. Test data, if available, can also be used to check a model. However, since test/analysis correlation is not an exact science, care should be taken in interpreting the results.

3.4 Checking Static Analysis Output

After a static analysis has been executed, NASTRAN provides several diagnostics that can be used to check the results. The output is described briefly in the following sections. Reference 6 contains a detailed description of these features.

3.4.1 FBS Diagnostics

MSC/NASTRAN solves static analysis problems by decomposing the stiffness matrix and then using forward/backward substitution (FBS) to solve for the displacement vectors. The FBS module provides useful diagnostics to help the user determine if there were numerical conditioning problems during the solution. First, a Residual Load Vector is calculated by subtracting the applied load vector from the product of the stiffness matrix times the calculated displacement vector:

$$\delta P = Ku - P$$

The Residual Load Vector is not printed unless the user requests it by inserting PARAM,IRES,1 in the Bulk Data Deck. A better measure of the error is obtained by computing the ratio of the work done by the residual forces to the work done by the applied forces:

$$\epsilon = \frac{u^T \delta P}{u^T P}$$

This error measure is printed under the heading "EPSILON." NASTRAN flags epsilons larger than 0.001, however MSC suggests that epsilons in the neighborhood of 10^{-9} are generally considered acceptable. The external work done by the applied loads is also reported.

3.4.2 OLOAD Resultant

The OLOAD Resultant is automatically calculated for each applied load vector. It represents the resultant of all applied loads referenced to the origin of the basic coordinate system (or to the grid point specified by PARAM,GRDPNT). Although this computation is

an applied loads check, and is not really a model check, it is an important consideration when static loads are being used to check out a model. OLOAD output at the grid point level can be requested by using the Case Control OLOAD card.

3.4.3 SPCFORCE Resultant

The SPCFORCE Resultant is the summation of all forces of single point constraint with respect to the reference point. As in the OLOAD summation, the reference point is either the grid point specified by PARAM,GRDPNT, or the origin of the basic coordinate system. SPCFORCES can also be printed for individual grid points. A useful equilibrium check can be made by summing the SPCFORCE Resultant and the OLOAD Resultant.

3.4.4 Maximum Load

NASTRAN automatically prints a summary of the maximum load in each direction (of the basic coordinate system) for each load vector. Care must be taken when reading this output. The maximum load may not occur at the same grid point for any of the six directions, and the table gives no information as to where the maximum loads occurred. This information can be useful as a "sanity" check.

3.4.5 Maximum Displacement

The table of maximum displacements is also printed for each of the six basic coordinate directions for each load vector. No location information is given, however the table is another useful means of checking that the magnitudes of the displacements make sense for each direction.

4.0 Model Verification Procedure

4.1 Introduction

The model verification procedure described in this section is based on the techniques outlined in the previous sections. The procedures have been used for checking a non-superelement High Speed Civil Transport (HSCT) model generated outside of LaRC, delivered in a foreign FE code format and translated into MSC/NASTRAN format. As described in Section 1, the procedures are intended primarily to verify that the model is mathematically correct. Modeling issues such as mesh density, element type and usage, and connection details are difficult to check in an objective sense, and are beyond the scope of this document. It is assumed in this section that the model has already been translated into MSC/NASTRAN form and has been checked with a graphical preprocessor. See the discussion on analytical checks in Section 3 for more detailed descriptions of the MSC/NASTRAN procedures and calculations.

4.2 Constraint Checks

Set up a static analysis run (SOL 101), and include "PARAM,CHECKOUT,YES" in the Case Control or Bulk Data deck. A sample deck is shown below.

```

NASTRAN BUFFSIZE=4096 SPARSE=25
$ NOTE: SPARSE=25 not needed for Version 68
INIT DBALL LOGI=(DBALL(500000))
SOL 101
TIME 300
DIAG 8,15
CEND
$
TITLE = MODEL CHECKOUT RUN
SUBTITLE = PARAM CHECKOUT YES
ECHO = NONE
MAXLINES = 200000
LINE = 52
$
PARAM,CHECKOUT,YES
$
BEGIN BULK
$ Include mass data - optional for this run
include 'mass1.bdf'
$ Parameters used in all static and dynamic runs
PARAM AUTOSPC YES
PARAM K6ROT 5.0
PARAM POST -2
PARAM NEWSEQ -1
PARAM MAXRATIO 1.E7
PARAM BAILOUT -1

```

Rest of Bulk Data

ENDDATA

Check the output for the following:

1. Look at the printout of matrix E_{mh} (EMH). Nonzero terms in EMH indicate improper multipoint constraint equations, i.e., equations having internal constraints. This could be caused by errors on MPC cards or rigid element cards such as RBARs. Note that the use of MSC/NASTRAN's rigid elements (RBARs, RBE2s, etc.) instead of MPCs wherever possible will help prevent multipoint constraint errors because the program automatically generates the constraint equations for these elements. If there are no internal constraint errors, all columns of EMH will be null.
2. Look for messages from the DCMP module. NASTRAN decomposes the matrices R_{mm}^g (RGMM) and R_{mm}^m (RMMM) to detect the presence of linearly dependent equations at both the G set level and the M set level. If there are errors of this type, NASTRAN will issue several error messages, including the following:
 - USER FATAL MESSAGE 6137 (rank deficient matrix)
 - USER FATAL MESSAGE 5225 (attempt to operate on a singular matrix)
 - USER INFORMATION MESSAGE 4158 (decomposition statistics)
 - USER INFORMATION MESSAGE 4698 (more decomposition statistics)

User Information Message 4698 is the most useful of these because it lists the grid point number and degree of freedom at which the error occurs. Errors in RGMM will propagate down to RMMM. However, an error in RMMM that does not appear in RGMM indicates that a change in the M set exists which will fix the problem (see discussion in section 3).

4.3 Multi-Level Strain Energy Checks

Set up another static analysis run. Include the DMAP Alter "checka.v68" for Version 68 (or "checka.v675" for Version 67.5) located in the "/misc/sssalter" directory (or [.misc.sssalter] on VMS machines). Set the parameters to the values shown in the following example.

```
SOL 101
TIME 300
DIAG 8,15
echooff
include 'checka.v675'
echoon
CEND
$
TITLE = MODEL CHECKS
SUBTITLE = MSC ALTERS
ECHO = NONE
$
MAXLINES = 200000
LINE = 52
$
$ **** Comment out SPC selection to check model in free-free condition
$ SPC = 1
BEGIN BULK
$ Parameters for multilevel mass and stiffness checking
$ 
$     CHKMASS = 1 => check mass matrices
$     CHKSTIF = 1 => check stiffness matrices
$ 
PARAM,CHKSTIF,1
PARAM,CHKMASS,1
$ Other analysis parameters
$PARAM AUTOSPC YES
PARAM K6ROT 5.0
$PARAM,POST,-2
PARAM,NEWSEQ,-1
PARAM, MAXRATIO, 1.E7
PARAM, BAILOUT, -1
PARAM, WTMASS, .00259
PARAM GRDPNT 5754
$ Include mass data (or use editor to include in bulk data)
include 'mass1.bdf'

.
.
.
Rest of Bulk Data
.
.
.

ENDDATA
```

Notes:

1. There are no output or analysis requests because this is a "pre-analysis" check run.
2. The Alter cannot be used with the "scr=yes" option. You must create a database. (The database files can be deleted after the run.)

Check the output for the following:

1. Check the Grid Point Weight Generator (GPWG) Output. Specific items to be checked include Mass, Center of Gravity, Inertia Matrix, I(s), and the Rigid Body Mass Properties Matrix, MO. (N.B.: The data is computed in weight units according to the value set on PARAM,WTMASS, and the properties are computed with respect to the reference point selected by PARAM,GRDPNT. If PARAM,GRDPNT is not specified, the reference point is taken as the origin of the Basic coordinate system.) Reference 6 contains a thorough discussion of the GPWG. In the example above grid point 5754 was chosen because it is near the CG.
2. At each set level (G-set, N-set, and A-set) check the strain energy matrix (CHKGG, CHKKNN, or CHKKAA). All diagonal terms should be very small. The DMAP will issue a warning message, if any of the diagonal terms are greater than 10^{-5} . Experience has shown that this message may be safely ignored if translational terms are less than about 10^{-3} and rotational terms are less than about 10^1 . Rotational terms are a function of the reference point used to compute the rigid body vectors. Choosing a reference point outside the structure, for example, may cause the rotational strain energy to exceed 10^1 .
3. If the diagonal terms of a strain energy matrix are not nearly zero (as defined above), inspect the reaction force matrices to determine the degree(s) of freedom causing the problem. The reaction force matrices (REACGNRM, REACNNRM and REACANRM) are normalized such that the largest force is 1.0.
4. Inspect the rigid-body mass matrices (WGHT, WGHTN and WGHTA), and compare them to the matrix, MO, output by the Grid Point Weight Generator. Possible reasons for discrepancies include improper dynamic reduction and errors in rigid body elements or MPCs. Some terms in the A-set matrix, WGHTA, may be less than the corresponding terms in MO, if boundary conditions (e.g., symmetric, antisymmetric) have been included in the calculations by specifying an SPC set in the Case Control section. It is a good idea to perform the checks for both the free-free and constrained conditions for half-models that use symmetric or antisymmetric boundary conditions to simulate the other half of the structure.

4.4 Static Analysis: 1-g Check

Set up a Solution 101 (Superelement Statics) run similar to the one shown below.

```
SOL 101
TIME 300
DIAG 8,15
CEND
$
TITLE = MODEL CHECKS
```

```

SUBTITLE = 1-G Loads in X, Y, and Z directions
ECHO    = NONE
$ 
$ 
MAXLINES = 200000
LINE = 52
$ 
DISP(plot)=all
$ 
$ ----- 
SUBCASE 1
LABEL = Gravity in +X Direction, Symmetric BCs
SPC   = 1
LOAD  = 101
SUBCASE 2
LABEL = Gravity in +Y Direction, Symmetric BCs
SPC   = 1
LOAD  = 102
SUBCASE 3
LABEL = Gravity in +Z Direction, Symmetric BCs
SPC   = 1
LOAD  = 103
BEGIN BULK
include 'mass1.bdf'
GRAV 101 0 386.1 1.0 0.0 0.0
GRAV 102 0 386.1 0.0 1.0 0.0
GRAV 103 0 386.1 0.0 0.0 1.0
$ 
PARAM AUTOSPC YES
PARAM K6ROT 5.0
PARAM POST -2
PARAM NEWSEQ -1
PARAM MAXRATIO 1.E7
PARAM BAILOUT -1
PARAM GRDPNT 0
PARAM WTMASS .00259
.
.
$ Constraints for Static Test Loads
$ -----
$ The following constraints are used to support the model for the
$ static load checks. They are used in addition to the free-free
$ symmetric constraints. They should be removed before calculating
$ symmetric normal modes.
$ 
SPC1      1      3      5639
SPC1      1     13      5754
.
.
.
ENDDATA

```

Notes:

1. The gravity loads are applied with symmetric boundary conditions enforced for the subcases shown (SPC=1). Additional supports (shown in the Bulk Data deck above)

are required to remove rigid-body motion in the free DOF. Since Version 68 of MSC/NASTRAN allows changes in SPC sets between subcases, the anti-symmetric boundary conditions could also be checked in the same run by adding three more subcases which refer to a different SPC set.

2. PARAM,GRDPNT turns on the Grid Point Weight Generator. Although this is optional, weight calculations are inexpensive, and the output provides one more item that can be used to verify that the input data is correct.

Check the output for the following:

1. Check the Grid Point Weight Generator output. Look for correct weight, C.G. and moments of inertia.
2. OLOAD RESULTANT: The resultant of forces in the translational directions for each subcase should be compared to the known weight of the structure. If the Grid Point Weight Generator used the origin of the basic coordinate system as a reference point (PARAM,GRDPNT,0), then the resultant (OLOAD) moments should be the same as the corresponding terms of the matrix, MO.
3. Inspect the messages output by the DCMP module (sparse decomposition). User Information Message 4158 provides statistics which inform the user of the numerical quality of the stiffness matrix, KLL.
4. User Information Message 5293, output by the forward/backward substitution (FBS) module, provides more information about the numerical quality of KLL. The error measure, epsilon, and the external work are printed.
5. Check the maximum displacements and applied loads. Remember that these are a function of (possibly arbitrary) boundary conditions.
6. Finally, a good visual confirmation of the results can be obtained by plotting the deformed shapes. Look for excessive local displacements or "kinks" that might indicate missing elements or constraints.

4.5 Static Analysis: Enforced Displacement Check

Set up a Solution 101 (Superelement Statics) run similar to the one shown below. Note that MSC/NASTRAN versions prior to version 68 do not allow boundary condition (SPC set) changes between subcases (except for SOL 24).

```
SOL 101
TIME 300
DIAG 8,15
CEND
$
TITLE = MODEL CHECKS
SUBTITLE = Enforced Displacements
ECHO = NONE
$
MAXLINES = 200000
```

```

LINE = 52
$DISP(PLOT) = ALL
$SPCFORCES = ALL
$ Model in Free-Free State
SUBCASE 1
  LABEL = X translation - free-free model
  SPC = 3
  LOAD = 100
SUBCASE 2
  LABEL = Y translation - free-free model
  SPC = 3
  LOAD = 200
SUBCASE 3
  LABEL = Z translation - free-free model
  SPC = 3
  LOAD = 300
SUBCASE 4
  LABEL = X rotation - free-free model
  SPC = 3
  LOAD = 400
SUBCASE 5
  LABEL = Y rotation - free-free model
  SPC = 3
  LOAD = 500
SUBCASE 6
  LABEL = Z rotation - free-free model
  SPC = 3
  LOAD = 600
$ Symmetric Boundary Conditions
SUBCASE 11
  LABEL = X translation - SYMMETRIC BCS
  SPC = 13
  LOAD = 100
SUBCASE 12
  LABEL = Z translation - SYMMETRIC BCS
  SPC = 13
  LOAD = 300
SUBCASE 13
  LABEL = Y rotation - SYMMETRIC BCS
  SPC = 13
  LOAD = 500
$ Antisymmetric Boundary Conditions
SUBCASE 21
  LABEL = Y translation - ANTISSYMMETRIC BCS
  SPC = 23
  LOAD = 200
SUBCASE 22
  LABEL = X rotation - ANTISSYMMETRIC BCS
  SPC = 23
  LOAD = 400
SUBCASE 23
  LABEL = Z rotation - ANTISSYMMETRIC BCS
  SPC = 23
  LOAD = 600
$
BEGIN BULK
$
PARAM AUTOSPC YES
PARAM K6ROT 5.0
PARAM POST -2
PARAM OGEOM NO
PARAM NEWSEQ -1
PARAM MAXRATIO 1.E7

```

```

PARAM  BAILOUT -1
$ -----
$ Data for Applying Enforced Displacements
$ -----
$ SPC SET      DEFINITION
$ -----
$   1          SYMMETRIC BC -- (CARDS LOCATED AT END OF BULK DATA)
$   2          ANTISSYMMETRIC BC --
$   3          CONSTRAIN GRID POINT 5754 (NEAR CG) IN ALL SIX DOF
$  13          COMBINE SET 1 AND SET 3
$  23          COMBINE SET 2 AND SET 3
$
SPC1    3      123456  5754
SPCADD  13     1      3
SPCADD  23     2      3
$
$ ENFORCED DISPLACEMENTS AT GRID POINT 5754 (CHOSEN
$ BECAUSE IT IS NEAR THE CG)
$ -----
SPCD    100    5754    1      1.0
SPCD    200    5754    2      1.0
SPCD    300    5754    3      1.0
SPCD    400    5754    4      1.0
SPCD    500    5754    5      1.0
SPCD    600    5754    6      1.0
$
$ -----
.
.
.
ENDDATA

```

Check the output for the following:

NOTE: NASTRAN will issue a series of warning messages (UWM 3204). Ignore these. They are left over from the older versions of MSC/NASTRAN which required a dummy load card with enforced displacements (e.g., SPCDs).

1. OLOAD RESULTANT: All terms should be zero since no loads were applied.
2. Inspect the messages output by the DCMP module (sparse decomposition). User Information Message 4158 provides statistics which inform the user of the numerical quality of the stiffness matrix, KLL.
3. User Information Message 5293, output by the forward/backward substitution (FBS) module provides more information about the numerical quality of KLL. The error measure, epsilon, and the external work are printed.
4. Check the spc force resultant, the maximum spc forces, maximum displacements and maximum applied loads. Note that the spc forces will not be exactly zero, especially for the rotational DOF.
5. Finally, a good visual confirmation of the results can be obtained by plotting the deformed shapes. The plots should not show any visible deformation other than rigid-body motion.

4.6 Static Analysis: Checks Against Reference Data

If data is available from the finite element model developer, a static analysis run can be set up to verify results such as displacements at key locations caused by prescribed forces. The same care should be taken to methodically check the output of such a run, however, the objective is to simply compare results. Since this procedure would vary depending on the model, no example is given here.

5.0 Summary

Finite element models (FEMs) are the basis for a variety of engineering computations such as stress and stability analysis, and vibration and dynamic load analysis. Although a carefully developed, thoroughly validated FEM is always desirable, it is of utmost importance when the model is being used in the preliminary design stages of a large project. Since no data is available to validate the model, FEM developers must use their best engineering judgment to model the structure accurately. FEM users may want to assess the suitability of the developer's modeling techniques, however a user's primary validation task is to ensure that the model is mathematically correct and does not contain any inadvertent errors. This document outlines a suggested procedure for accomplishing this task.

6.0 References

1. "PATRAN Plus User Manual," PDA Engineering, October 1989.
2. "MSC/NASTRAN Reference Manual, Version 68," Lahey, R.S., Miller, M.P. and Reymond, M., The MacNeal-Schwendler Corporation, 1994.
3. "I-DEAS Master Series," Structural Dynamics Research Corporation, 1993.
4. "Development and Applications of a Multi-Level Strain Energy Method For Detecting Finite Element Modeling Errors", Hashemi-Kia, M., Kilroy, K., and Parker, G., NASA CR 187447, October 1990.
5. "Diagnostics in Finite Element Analysis," Haggenmacher, G.W. and Lahey, R.S., First Chautauqua on Finite Element Modeling, September 15-17, 1980.
6. "MSC/NASTRAN Linear Static Analysis User's Guide, Version 68," Caffrey, J.P. and Lee, J.M., The MacNeal-Schwendler Corporation, 1994.
7. "MSC/NASTRAN Programmer's Manual, Version 64," The MacNeal-Schwendler Corporation, 1986.
8. "MSC/NASTRAN Numerical Methods User's Guide, Version 67," Komzsik, L., The MacNeal-Schwendler Corporation, 1992.
9. "MSC/NASTRAN Handbook for Numerical Methods, Version 66," Komzsik, L., The MacNeal-Schwendler Corporation, 1990.



REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188
<p>Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.</p>			
1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED	
	June 1995	Contractor Report	
4. TITLE AND SUBTITLE	5. FUNDING NUMBERS		
A Verification Procedure for MSC/NASTRAN Finite Element Models	C NAS1-19000 WU 233-01-01-03		
6. AUTHOR(S)			
Alan E. Stockwell			
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)	8. PERFORMING ORGANIZATION REPORT NUMBER		
Lockheed Engineering and Sciences Company Langley Program Office 144 Research Drive Hampton, VA 23666			
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSORING / MONITORING AGENCY REPORT NUMBER		
National Aeronautics and Space Administration Langley Research Center Hampton, VA 23681-0001	NASA CR-4675		
11. SUPPLEMENTARY NOTES			
Langley Technical Monitor: Howard M. Adelman			
12a. DISTRIBUTION / AVAILABILITY STATEMENT	12b. DISTRIBUTION CODE		
Unclassified - Unlimited Subject Category 05			
Availability: NASA CASI, (301) 621-0390			
13. ABSTRACT (Maximum 200 words)			
<p>Finite element models (FEMs) are used in the design and analysis of aircraft to mathematically describe the airframe structure for such diverse tasks as flutter analysis and actively controlled landing gear design. FEMs are used to model the entire airplane as well as airframe components. The purpose of this document is to describe recommended methods for verifying the quality of the FEMs and to specify a step-by-step procedure to implementing the methods.</p>			
14. SUBJECT TERMS			15. NUMBER OF PAGES
Finite element; modeling; verification; MSC/NASTRAN			24
			16. PRICE CODE
			A03
17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	

